

CERTIFICATION

This is to certify that

Vittorio Loggia

has successfully completed a course

Learn Altium Essentials

Date: 01 MAR 2021 - 11 APR 2021

Course tutor

Robert Teranec



Software used: **Altium Designer**

Course content: 1) Starting a new project, Creating a schematic symbol library 2) Drawing a schematic 3) Creating footprints and a PCB library 4) Doing PCB Layout 5) Improving PCB Layout 6) Finishing PCB, Generating manufacturing outputs

Certificate number: 1c651a67904c2a7f7700473f0